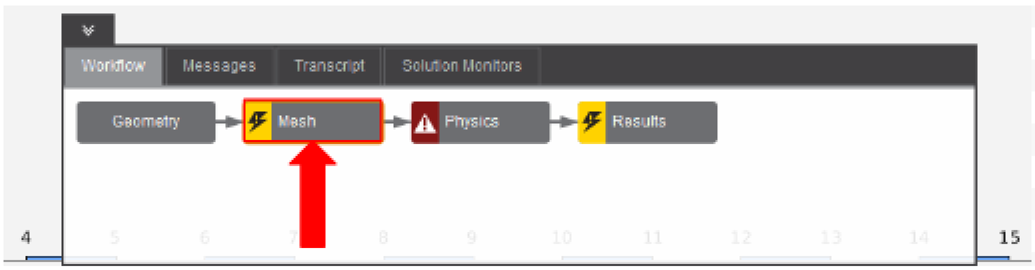
ANSYS Flow in a S-Duct - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification
1. Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Results
6. Verification & Validation

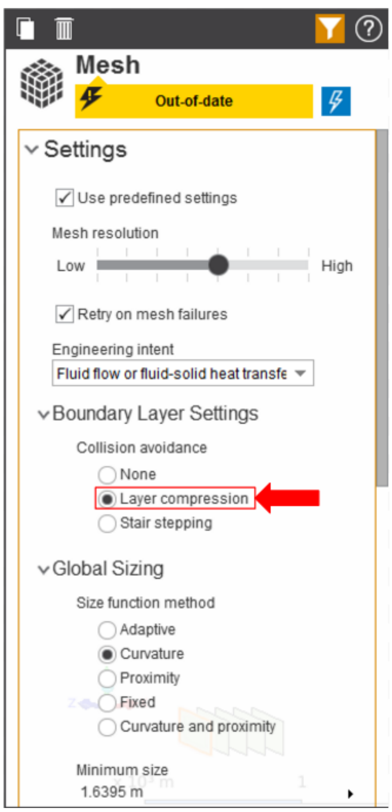
Mesh

Initiate the meshing process by clicking on **Mesh** in the workflow.



Set Mesh Boundary Layer

In the **Boundary Layer Settings**, under **Collision avoidance**, select **Layer compression**.



AIM will prompt you to fix the **Boundary Layer** before generating the mesh. Click on **Boundary Layer** under **Mesh Controls**. Select every face except for the inlet and outlet faces (133 faces total).

Generate Mesh

Return to the **Mesh** panel, then click **Generate Mesh** under **Output** or at the top of the screen by the status window for **Mesh**. AIM will detect that you are ready to generate the mesh and highlight the buttons in blue.

Go to Step 4: Physics Setup

Go to all ANSYS AIM Learning Modules